
Computational Fluid Dynamics Uses in Fluid Dynamics/ Aerodynamics Education

Terry L. Holst, Ames Research Center, Moffett Field, California

July 1994



Ames Research Center
Moffett Field, California 94035-1000

Computational Fluid Dynamics Uses In Fluid Dynamics/Aerodynamics Education

TERRY L. HOLST

Ames Research Center

Summary

The field of computational fluid dynamics (CFD) has advanced to the point where it can now be used for the purpose of fluid dynamics physics education. Because of the tremendous wealth of information available from numerical simulation, certain fundamental concepts can be efficiently communicated using an interactive graphical interrogation of the appropriate numerical simulation data base. In other situations, a large amount of aerodynamic information can be communicated to the student by interactive use of simple CFD tools on a workstation or even in a personal computer environment. The emphasis in this presentation is to discuss ideas for how this process might be implemented. Specific examples, taken from previous publications, will be used to highlight the presentation.

Introduction

Use of computational fluid dynamics (CFD) tools by the aerospace field to increase understanding of fluid dynamic and aerodynamic phenomena has been rapidly increasing during the past decade especially the last several years. The primary reasons for this are the rapidly increasing simulation capabilities in the CFD field and the rapidly expanding capabilities in computer hardware performance. For example, computer hardware execution speed has increased by a factor of about 15 over the past decade and by over 200 during the past two decades. This rapid advance in computational execution speed is displayed in figure 1. The top curve shows how the theoretical peak execution speed has improved with time and includes effects from both circuit speed and architectural improvements, e.g., vectorization speed ups from Cray-type computers. The lower curve represents the improvement in execution speed due to just circuit speed. The middle curve represents the actual improvement in execution speed performance from a variety of CFD application codes as approximated from the shaded symbols.

Another reason for the dramatic increase in the use of scientific computational tools is that industry has discovered the positive influence that computational analysis can

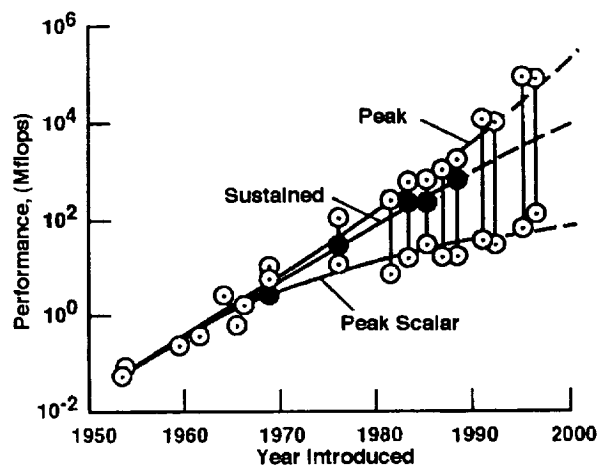


Figure 1. Conventional main-frame computer execution speed improvement as a function of time, results taken from references 1-4.

have on aircraft, spacecraft, and missile design. Improved efficiency in aerospace vehicle performance at reduced design cost and risk is a direct result of increased use of computational simulations. Indeed, additional advancement in this area is crucial to enable the United States to maintain its technological advantage in the aerospace sciences.

Just as numerical simulation has become a significant and growing aspect of the aircraft design process, the stage is set for a dramatic increase in the utilization of CFD in the educational arena. In this context, it is not meant to imply that the study of CFD will increase dramatically, but that the use of CFD as a teaching tool for other areas or disciplines of fluid dynamics will increase. This utilization should range from enhancing the understanding of nonlinear engineering models, e.g., the aerodynamics of transonic wings, to obtaining a better understanding of fluid physics, e.g., flat plate boundary layer transition. Through the synergistic utilization of CFD coupled with an appropriate level of experimental validation, students will obtain a better understanding of the physical aspects of aerodynamics and fluid mechanics as well as how to

interpret the effects of numerical error associated with CFD solutions.

The remainder of this paper will provide a review of some of the current areas of CFD research that may be utilized in the educational environment in the near future. These areas are especially attractive if improvements in workstation computing continues at today's current fast rate. In addition, areas that may be used more immediately, i.e., even today, will be presented and discussed.

Review of CFD Applications

The first results used to establish the abilities of CFD are a set of full potential solutions for a variety of transonic wing configurations (taken from refs. 5–7). In these simulations the nonlinear full potential equation is solved for the inviscid transonic flow field complete with transonic shock waves. Results from two different full potential computer codes, TWING (ref. 5) and FLO28 (ref. 6), are compared with experiment (ref. 7) in figure 2 for the ONERA M6 wing at transonic flow conditions. Although there are some discrepancies in the computed results, both show the same trends, i.e., they both predict a double shock structure on the upper surface including a supersonic-to-supersonic oblique shock swept

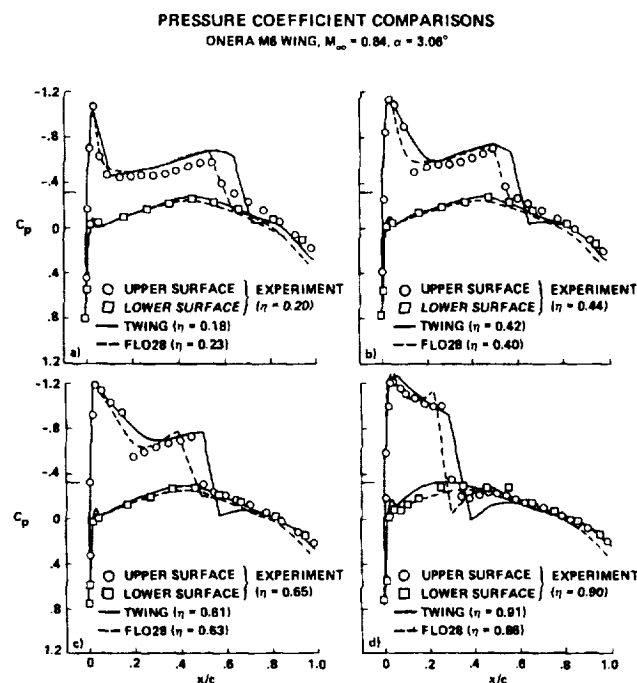


Figure 2. Pressure coefficient comparisons at four semi-span stations on an ONERA M6 wing, $M_\infty = 0.84$, $\alpha = 3.06$ deg (ref. 5).

approximately parallel to the wing leading edge. Most of the discrepancies are a direct result of a coarse grid used in the numerical simulations, a direct result of main memory limitations from a decade ago.

An additional full potential result computed with the TWING code and taken from reference 8 is shown in figure 3. In this figure the drag-rise characteristics (C_D versus M_∞) are compared for two transonic wing cases: an original or baseline wing and an optimized wing. The baseline geometry was modified using the QNMDIF optimization code (refs. 9–10) to produce the optimized wing by minimizing the value of cruise D/L. As can be seen from figure 3 the drag-rise characteristics of the optimized wing are significantly improved over the original baseline wing. It should be pointed out that the drag values associated with figure 3 are pressure drag values only, i.e., they do not contain skin friction drag.

The most interesting aspect of these simulations, especially in the present context, is the amount of computer time required for a complete simulation. The computing times reported in reference 5 are on the order of 10–20 sec on a single processor of a Cray Y-MP computer. An entire aerodynamic performance curve, such as the drag rise curve presented herein, requires on the order of only one minute. This machine was in the supercomputer class just a decade ago but now is about even with a high end workstation in execution speed and does not even match a reasonably advanced personal computer in terms of main memory. Such simulations would be easily adapted to the educational environment and will be discussed in more detail in the next section of this paper.

The next example results used to establish the state of the art in CFD applications are a set of Reynolds-averaged

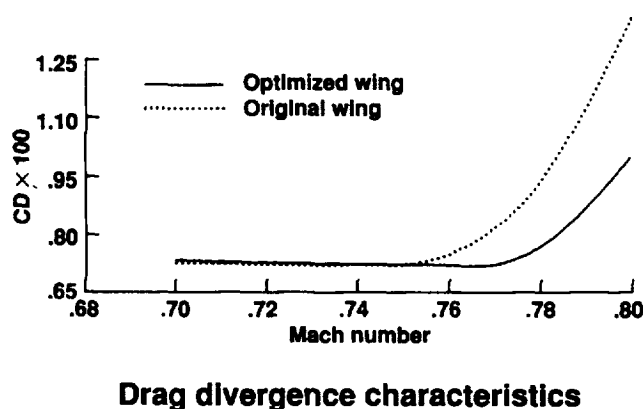


Figure 3. Coefficient of drag ($\times 100$) versus Mach number for baseline and optimized wings (ref. 8).

Navier–Stokes (RANS) simulations about a canard–wing–fuselage configuration (refs. 11–13). The geometry consists of an ogive-cylinder fuselage with a canard and wing composed of circular-arc airfoil sections. The canard and wing are closely coupled, have zero-twist, are mid-mounted, and are highly swept and tapered. Precise geometric details can be found in references 11–13.

Numerically computed results compared with experimental results taken from reference 14 are presented in figures 4 and 5. Figure 4 shows a variety of force and moment comparisons for the canard–wing–fuselage configuration for both deflected canard (10 deg) and undeflected canard cases over a range of angles of attack. Note the generally good agreement with experiment for these comparisons. Figure 5 shows comparisons of component lift and pitching moment for the deflected canard case. Again the computed results are in good agreement with experiment. For this set of computations there is a complex canard–wing vortex interaction that exists, especially for the higher angles of attack. Some cases have even been computed that predict wing and canard vortex breakdown, a phenomenon that is also present in the experiment.

The results presented in figures 4 and 5 demonstrate the ability of RANS methods to predict the aerodynamic characteristics of reasonably complex configurations even in the presence of significant viscous effects. Such computations would not have been possible with the full potential approach or even an Euler equation approach. The computer time expense associated with these RANS computations is considerably larger than for the full potential computations presented previously and ranges from about 2 to 15 hours of single processor time on a Cray C-90 supercomputer. The variation in required execution time is due to variations in geometry, flow conditions, and grid refinement level. Complete details of this set of computations are given in references 11–13.

The next example simulation involves the numerical solution of the Reynolds-averaged Navier–Stokes (RANS) equations for the flow field about a nearly complete F-18 aircraft. This effort was accomplished with a series of flow solutions about several increasingly complete geometrical representations of the F-18. This series started with forebody flow solutions (refs. 15–17), proceeded with wing–fuselage simulations (refs. 18–19), and to date has concluded with a nearly complete F-18 aircraft that includes the fuselage, tail, and wing with deflected leading edge flap (ref. 20). Results from the last effort utilizing the most complete geometry are presented in figures 6 and 7. This level of geometric complexity is possible in a structured grid approach because of the zonal grid scheme used. The flow field is broken into a series of grid zones

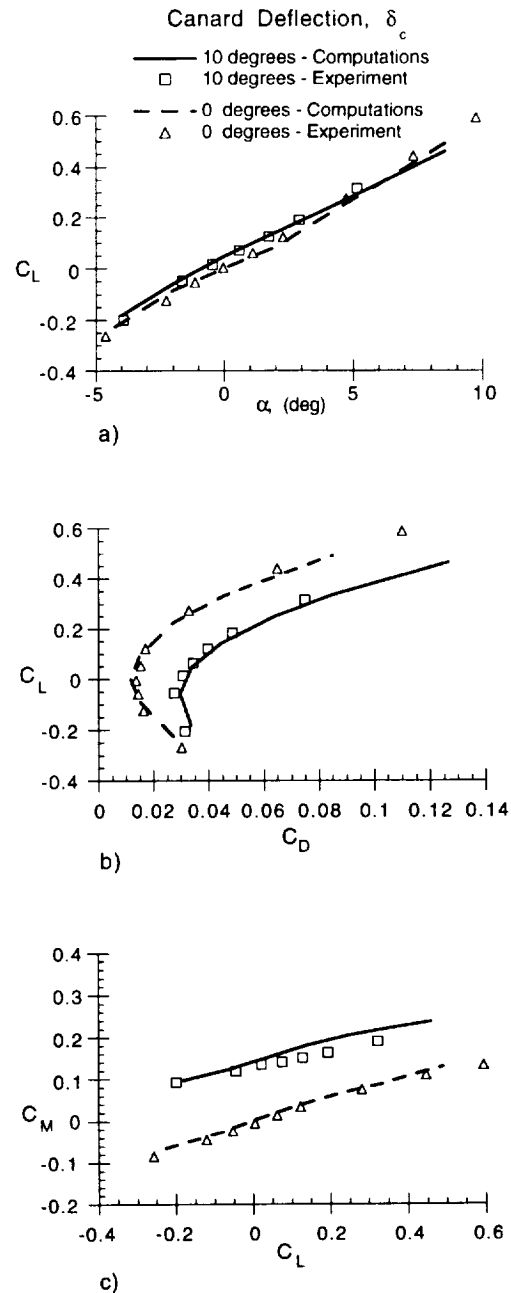


Figure 4. Comparison of computed and experimental forces and moments for a canard–wing–body configuration for both undeflected and deflected canard cases, $M_\infty = 0.84$. (a) Lift curves, (b) drag polars, (c) moment curves.

using an overset or chimera zonal grid approach. Each grid zone is designed to capture the effects of one aspect of the geometry, e.g., vertical tail, wing, or the leading-edge flap, and is generated without regard to the other

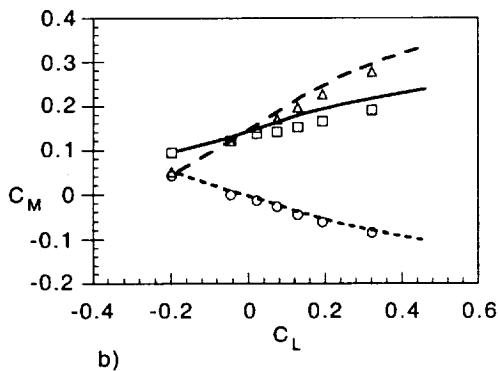
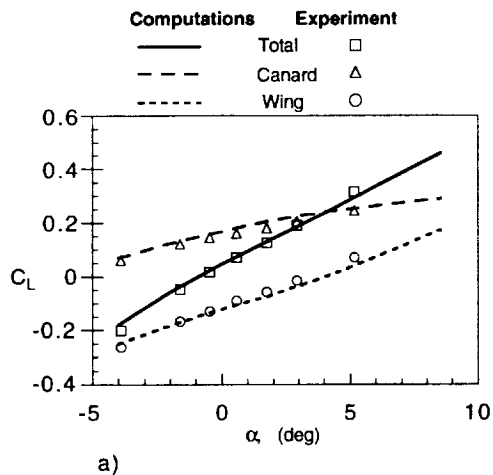


Figure 5. Comparison of component lift and pitching moment curves for a canard-wing-body configuration in which the canard is deflected at 10 deg, $M_\infty = 0.84$. (a) Lift curves, (b) moment curves.

geometrical aspects of the overall configuration. During the flow solution process, information from one grid zone is communicated to another zone using a general interpolation procedure.

The present F-18 simulation presented in figures 6 and 7 utilized 10 chimera grid zones in conjunction with a bilateral plane of symmetry, which resulted in a grid of about 900,000 points. The simulation was run for 1,000 iterations in a non-time-accurate mode and an additional 4,100 iterations in a time-accurate mode. The solution required 8 million words of memory on a Cray Y-MP computer and 55 hours of single-processor CPU time. Figure 6 shows the limiting streamlines that have been computed just off the aircraft surface, and figure 7 shows three-dimensional particle traces and limiting

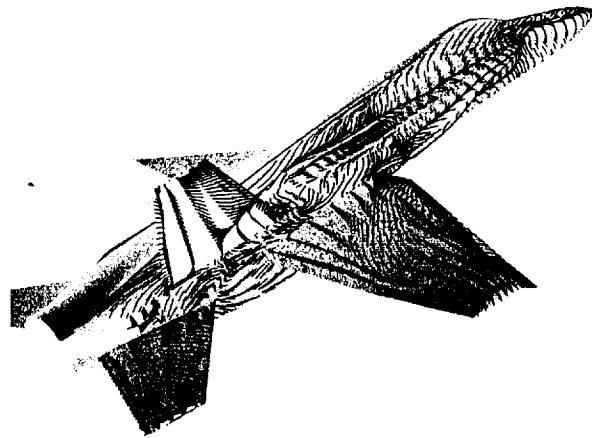


Figure 6. Computed limiting surface streamlines, F-18 aircraft, $M_\infty = 0.243$, $\alpha = 30.3$ deg, $Re_L = 11 \times 10^6$ (ref. 25).

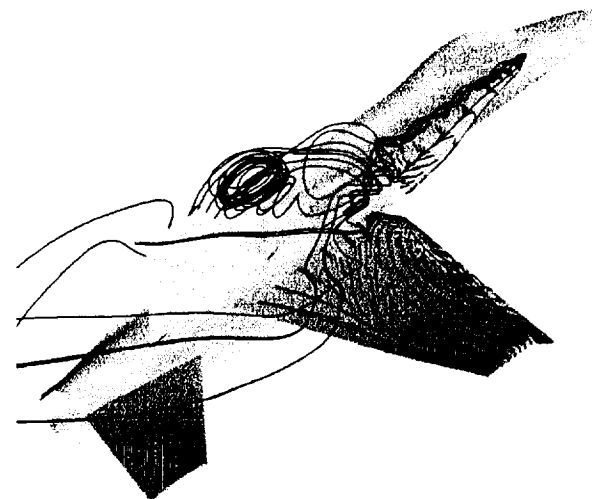


Figure 7. Computed particle traces showing LEX vortex breakdown and forward surface flow pattern, F-18 aircraft, $M_\infty = 0.243$, $\alpha = 30.3$ deg, $Re_L = 11 \times 10^6$ (ref. 25).

streamlines on the leading-edge extension (LEX), wing, and deflected leading-edge flap. Each of these figures displays only an instant in time for this time-dependent solution.

Zonal grid approaches of several different varieties have been used in a number of other applications to solve the RANS equations including Buning et al. (refs. 21–22) for ascent-mode Space Shuttle computations, Meakin and Suhs (ref. 23) and Dougherty et al. (ref. 24) for store-separation computations, Kiris et al. (ref. 25) for biofluid applications, Flores and Chaderjian (refs. 26–28) for a simulation of a reasonably-complete F-16A aircraft, and

Chawla et al. (ref. 29) and Smith et al. (ref. 30) for several powered-lift flow simulations.

An example of a powered lift-flow computation, taken from Smith et al. (ref. 31), is displayed in figure 8. In this figure the viscous flow about a Harrier YAV-8B aircraft in ground effect is displayed via a series of particle traces emanating primarily from the front and rear nozzles of the aircraft. The flow conditions are hover-like involving a low forward speed of 30 knots at an altitude of 30 feet above the ground plane. This computation involved 2.8 million grid points distributed within 18 chimera grid zones and about 40 CPU hours on a Cray Y-MP computer.

As can be seen from figure 8, there are many flow features that exist in this computation including a ground horseshoe vortex generated by the interaction of the

freestream and jet flows in ground effect. Other aspects that can be studied by analyzing the numerical data base include the fountain effect created by two parallel jets impinging side-by-side on the ground plane, heating on the aft fuselage caused by the aft hot jet fountain effect, the dramatic loss of lift experienced by powered-lift aircraft in ground effect (the so-called suck-down effect), and propulsion efficiency loss due to hot gas ingestion into the propulsion system inlets. Understanding all these complex flow phenomena, even in a qualitative sense, is a difficult task. Such a simulation would be very difficult to perform in today's educational environment due to the computational expense. But an interactive interrogation of such a "canned" solution file would yield a wealth of information about a state-of-the-art aerodynamics problem. More discussion on this idea will be presented in the next section of this paper.

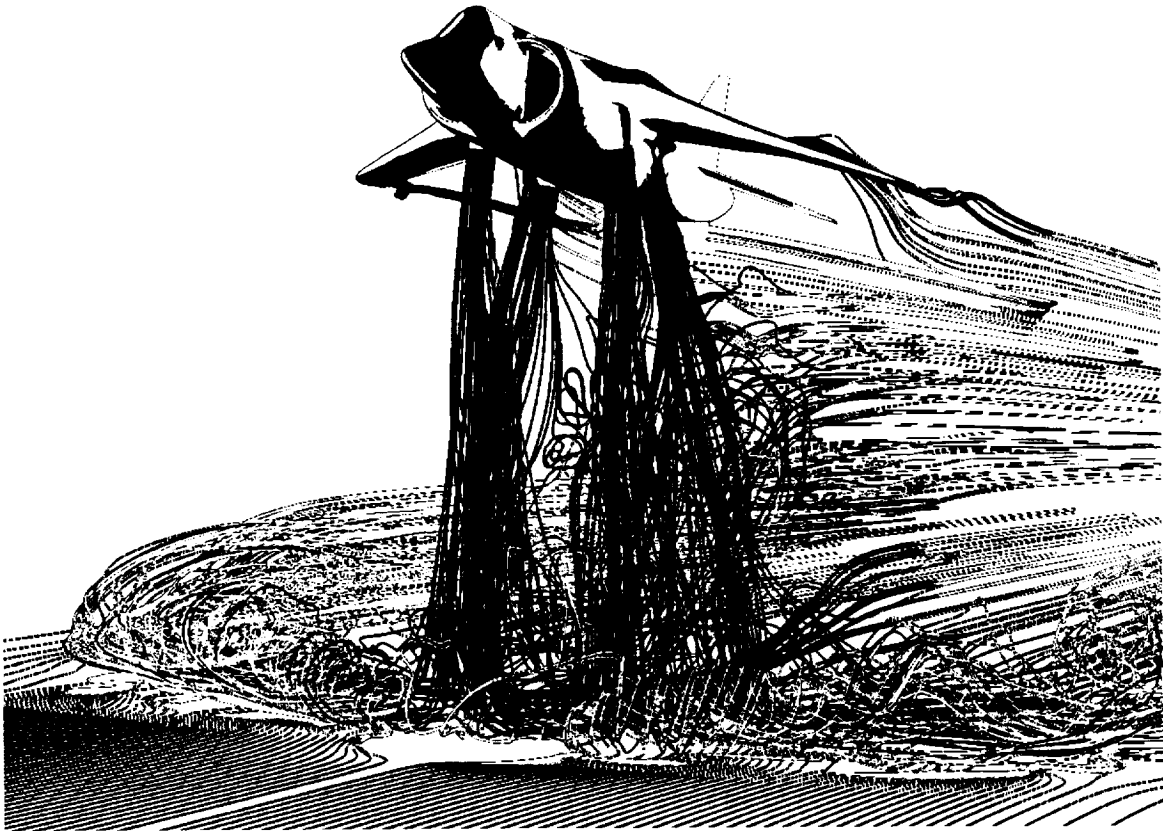


Figure 8. Numerically computed particle traces around a Harrier YAV-8B in ground effect at a speed of 30 knots and an altitude of 30 feet above the ground plane, $\alpha = 8.0$ deg (ref. 31).

Educational Uses of CFD

There are numerous reasons for utilizing computer simulations of fluid flows in the educational process. Developing hands-on experience, the power of parametric variation, the ability to easily optimize results, and the ability to visualize complete unsteady flow fields, are just a few capabilities that are either available now for the educational environment or will be available in the near future. The chief pacing items for further distribution of these capabilities are hardware cost effectiveness and suitable educationally oriented applications software availability. The hardware cost effectiveness, which is closely related to hardware speed, is rapidly improving as shown in figure 1. Another aspect of rapid computer efficiency improvement that is not shown in figure 1 is the rapid advance for "workstation" class computer hardware, i.e., desktop computers usually with a significant graphics capability. The capabilities associated with this class of computers has been climbing even more rapidly than that of the mainframe machines described in figure 1; it is this class of machines that have the capability to impact the educational environment most significantly.

Current high-end workstations can execute applications CFD software at 50 MFLOPS and higher. The cost-performance ratio of these machines may exceed that of large mainframe computers by as much as an order of magnitude. Although workstation speed is not fast enough to provide reasonable turn-around time for large applications, it is fast enough for many smaller applications, e.g., inviscid wing analysis or viscous airfoil analysis.^a Such computations typically require from just a few seconds to a few tens of minutes on such a high-end workstation. Of course, the precise amount of computer time intimately depends on the size of the grid used in the simulation and the exact formulation used. The uses of simulations in the educational environment can be categorized into three specific areas: (1) parametric analysis, (2) design (learning by trial and error), and (3) visualization. Each of these areas will now be discussed in more detail.

Parametric Analysis

Computational tools can be used parametrically to demonstrate a variety of trends that exist in the field of fluid mechanics, e.g., the linear relationship between lift and angle of attack for an airfoil or the dependence of a

^aAlthough many high-end workstation speeds are an order of magnitude or more slower than top of the line mainframe computers, the fact that they can be dedicated to one job or one user for large periods of time may, under the right circumstances, make them very effective for large applications as well.

flat plate laminar boundary layer thickness on the square root of Reynolds number. Traditionally, many of these fluid mechanics features are presented in the classroom in the form of theory and experiment, which are very valuable indeed. The intent of the present paper is not to advocate changing these established approaches, but rather to enhance them with a third alternative—CFD simulations. Each area, experiment, theory, and CFD, adds something a little different to the overall understanding of fluid mechanics. Thus, when all three are used together a combined synergistic effect is created. One aspect that the element of CFD brings to the overall understanding is parametric flexibility. What happens when compressibility is added to the problem or the C_{Lmax} point on a C_L versus α curve is reached? A CFD result can help answer these questions; it helps add to the explanation of a discrepancy between theory and experiment in complex non-linear flow regimes.

Computational simulations provide the potential to accommodate a broad range of applications with a broad range of parametric values. Applications software has been demonstrated today that spans the complete spectrum of flow situations including traditional incompressible, compressible subsonic, transonic, and supersonic flows about a variety of aircraft components and reasonably complete configurations; rarefied atmospheric entry flows (refs. 32–41); chemically reacting non-ideal-gas flows (refs. 42–45); flows involving combustion (refs. 46–49); flows over a wide range of Mach numbers and Reynolds numbers; and a large variety of internal flows (refs. 50–55). Providing hands-on experience with capabilities in a large variety of the above areas through experimental means would be prohibitively expensive for the educational environment even for the simplest of configurations or problems. Although computational results for some of these applications are expensive now, the future for cost effective computations in many of the above areas, especially for simplified geometry applications, is only a matter of time.

Design (Learning by Trial and Error)

The second major category of this section is actually a specialized subset of the previous section on parametric variation. In this case, the parameter being varied is the geometric shape. This type of problem is extremely important in aerodynamics and fluid dynamics, and thus, is given special treatment.

The key aspect of this area is to produce a desired effect or design, e.g., a specific cruise L/D for an airfoil or to maximize an airfoil's cruise L/D , by a parametric variation of the airfoil geometry. Examples of suitable parameters for this type of exercise include airfoil camber,

thickness distribution, angle of attack, leading-edge radius, trailing-edge angle, etc. The addition of constraints such as minimum wing volume, maximum wing root bending moment, and maximum adverse pressure gradient can be used to emphasize additional points of importance. Demonstrating the effects of some or all of these parameters and constraints on airfoil (or wing) L/D performance by using a CFD computer program can be informative and motivational. Students can be further motivated by having design competitions, i.e., who can come up with the highest cruise L/D for an airfoil subject to a specific set of constraints. Additional sophistication such as multiple design points, laminar flow control, or high-lift devices can dramatically enhance the teaching aid and extend the range of application. A design application, as described above, takes advantage of the ability of computational simulation to rapidly change geometric shape—a characteristic that would be difficult, if not impossible, to implement in an exclusively experimental approach.

Visualization

The third and final area suitable for impact by a CFD technique in the educational environment is that of visualization. Many fluid flow phenomena can be accurately simulated using CFD, but not in a short amount of computer time or wall clock time. Examples of such flows include the direct numerical simulation of the transitional flow over a flat plate or the simulation of flow about a complete aerospace vehicle such as those simulations presented at the beginning of this paper. The ability to perform such simulations in the educational environment will not be possible for many years to come.

However, the results of such simulations can be utilized in the classroom or student laboratory environment by using “canned” simulations, i.e., solution files that have been previously computed and saved on disk or some sort of CDROM device with a large storage capacity. Both computational and experimental results can be saved and recalled using this approach. These viewing sessions can be canned themselves, much like a movie film, or they can be interactive in nature with the student controlling the features to be analyzed. The latter approach would probably be the more stimulating and appealing. The results would be analyzed for pertinent details using the interactive graphical workstation environment. Direct comparisons between experiment and computation could be made. Most interactive sessions of this type would have to be for three-dimensional steady flows or two-dimensional unsteady flows, at least at first. Three-dimensional unsteady flows would require too much stor-

age space, and the interactive process would be too slow to achieve the proper characteristics within reasonable costs. With further improvements in the highly competitive graphical workstation market, additional improvements in hardware and system software may make even the largest simulations amenable to interactive analysis in the classroom environment.

Concluding Remarks

A brief review of CFD technology across a broad spectrum of capabilities, ranging from single-geometry full-potential simulations to complete aircraft Navier–Stokes simulations has been presented. The field of computational simulations is expected to continue rapid growth, with multidisciplinary applications significantly gaining popularity in the next decade. This trend will be especially true in the commercial airplane design environment.

This presentation has been made in the context of having an impact in the educational environment. Several areas are seen to have potential in this regard: (1) parametric analysis, (2) design (learning by trial and error), and (3) visualization. Overall the use of computer simulations and the interaction that computers allow is seen to be an important mechanism to enhance communication, motivation, and, ultimately, understanding in the educational environment.

References

1. Holst, T. L.; Salas, M. D.; and Claus, R. W.: The NASA Computational Aerosciences Program—Toward Teraflops Computing. AIAA Paper 92-0558, Jan. 1992.
2. Chapman, D. R.: Computational Aerodynamics Development and Outlook. AIAA J., vol. 17, no. 12, Dec. 1979, pp. 1293–1313.
3. Ballhaus, W. F., Jr.: Computational Aerodynamics and Supercomputers. Proceedings of the COMPCON '84 28th IEEE Computer Society International Conference, IEEE Computer Society, Silver Spring, Md., Feb.–Mar. 1984.
4. Kutler, P.; Steger, J. L.; and Bailey, F. R.: Status of Computational Fluid Dynamics in the United States. AIAA Paper 87-1135-CP, June 1987.
5. Holst, T. L.; and Thomas, S. D.: Numerical Solution of Transonic Wing Flowfields. AIAA J., vol. 21, no. 6, June 1983, pp. 863–870.

6. Caughey, D. A.; and Jameson, A.: Numerical Calculation of Transonic Potential Flow about Wing-Body Combinations. AIAA J., vol. 17, no. 2, Feb. 1979, pp. 175-181.
7. Schnitt, V.; and Charpin, F.: Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers. AGARD Report AR-138, May 1979.
8. Cosentino, G. B.; and Holst, T. L.: Numerical Optimization Design of Advanced Transonic Wing Configurations. AIAA J., vol. 23, no. 3, March 1986, pp. 192-199.
9. Gill, P. E.; Murray, W.; and Pitfield, R. A.: The Implementation of Two Revised Quasi-Newton Algorithms for Unconstrained Optimization. Division of Numerical Analysis and Computing, National Physical Lab Report NAC 11, 1972.
10. Gill, P. E.; and Murray, W. N.: Quasi-Newton Methods for Unconstrained Optimization. J. Inst. Math. and Appl., vol. 9, 1972, pp. 91-108.
11. Tu, E. L.: Navier-Stokes Simulation of a Close-Coupled Canard-Wing-Body Configuration. AIAA J., vol. 29, no. 5, Sept.-Oct. 1992, pp. 830-838.
12. Tu, E. L.: Effect of Canard Deflection on Close-Coupled Canard-Wing-Body Aerodynamics. AIAA J., vol. 31, no. 1, Jan.-Feb. 1994, pp. 138-145.
13. Tu, E. L.; Obayashi, S.; and Guruswamy, G. P.: Unsteady Navier-Stokes Simulation of a Canard-Wing-Body Ramp Motion. Presented at the AIAA 24th Fluid Dynamics Conference, AIAA Paper 93-3058, Orlando, Fla., July 1993.
14. Gloss, B. B.; Ray, E. J.; and Washburn, K. E.: Effect of Canard Vertical Location, Size, and Deflection on Canard-Wing Interference at Subsonic Speeds. NASA TM-78790, Dec. 1978.
15. Schiff, L. B.; Cummings, R. M.; Sorenson, R. L.; and Rizk, Y. M.: Numerical Simulation of High Incidence Flow over the F-18 Fuselage Forebody. AIAA Paper 89-0339, Jan. 1989.
16. Ghaffari, F.; Luckring, J.; Thomas, J.; and Bates, B.: Navier-Stokes Solution About the F/A-18 Forebody-LEX Configuration. AIAA Paper 89-0338, Jan. 1989.
17. Cummings, R. M.; Rizk, Y. M.; Schiff, L. B.; and Chaderjian, N. M.: Navier-Stokes Prediction of the Flow Field Around the F-18 (HARV) Wing and Fuselage at Large Incidence. AIAA Paper 90-0099, Jan. 1990.
18. Luckring, J. M.; and Ghaffari, F.: Status of Navier-Stokes Computations About the F/A-18 with Structured Grids. NASA High Angle of Attack Technology Conference, Hampton, Va., Oct. 31-Nov. 1, 1990.
19. Rizk, Y. M.; Schiff, L. B.; and Gee, K.: Numerical Simulation of the Viscous Flow Around a Simplified F/A-18 at High Angle of Attack. AIAA Paper 90-2999, Aug. 1990.
20. Rizk, Y. M.; and Gee, K.: Numerical Prediction of the Unsteady Flowfield Around the F-18 Aircraft at Large Incidence. AIAA Paper 91-0020, Jan. 1991.
21. Buning, P. G.; Chiu, I. T.; Obayashi, S.; Rizk, Y. M.; and Steger, J. L.: Numerical Simulation of the Integrated Space Shuttle Vehicle in Ascent. AIAA Paper 88-4359, Aug. 1988.
22. Buning, P. G. et al.: Flowfield Simulation of the Space Shuttle Vehicle in Ascent. Proceedings of Supercomputing '89: Supercomputer Applications, Fourth International Conference on Supercomputing and Third World Supercomputer Exhibition, Santa Clara, Calif., April 30-May 5, 1989.
23. Meakin, R. L.; and Suhs, N.: Unsteady Aerodynamic Simulation of Multiple Bodies in Relative Motion. AIAA Paper 89-1996-CP, June 1989.
24. Dougherty, F. C.; Benek, J. A.; and Steger, J. L.: On Application of Chimera Grid Scheme to Store Separation. NASA TM-88193, Oct. 1985.
25. Kiris, C.; Chang, I. D.; Kwak, D.; and Rogers, S. E.: Numerical Simulation of the Incompressible Internal Flow Through a Tilting Disk Valve. AIAA Paper 90-0682, Jan. 1990.
26. Flores, J.; and Chaderjian, N. M.: Zonal Navier-Stokes Methodology for Flow Simulation About a Complete Aircraft. AIAA J., vol. 27, no. 7, July 1990, pp. 583-590.

27. Flores, J.; and Chaderjian, N. M.: The Numerical Simulation of Transonic Separated Flow About the Complete F-16A. AIAA Paper 88-2506, June 1988.
28. Chaderjian, N. M.; and Flores, J.: Transonic Viscous Flow Computations About a Complete Aircraft Using the Navier-Stokes Equations. Presented at the 10th Australasian Fluid Mechanics Conference, Melbourne, Australia, Dec. 11-15, 1989.
29. Chawla, K.; Van Dalsem, W. R.; and Rao, K. V.: Numerical Study of a Delta Planform with Multiple Jets in Ground Effect. SAE 1989 Transactions, J. Aerospace, sec. 1, vol. 98, 1989, pp. 1555-1567.
30. Smith, M. H.; Chawla, K.; and Van Dalsem W. R.: Numerical Simulation of a Complete STOVL Aircraft in Ground Effect. AIAA Paper 91-3293-CP, Sept. 1991.
31. Smith, M. H.; Chawla, K.; and Van Dalsem W. R.: Numerical Simulation of a Complete STOVL Aircraft in Ground Effect. Presented at the AIAA International Powered Lift Conference, Santa Clara, Calif., AIAA Paper 93-4880, Dec. 1993.
32. Bird, G. A.: Molecular Gas Dynamics, Clarendon Press, Oxford, 1976.
33. Bird, G. A.: Monte Carlo Simulation in an Engineering Context, S. Fisher, ed., AIAA Progress in Astronautics and Aeronautics: Rarefied Gas Dynamics, vol. 74, part 1, 1981, pp. 235-239.
34. Bird, G. A.: Application of the Direct Simulation Monte Carlo Method to the Full Shuttle Geometry. AIAA Paper 90-1692, June 1990.
35. Baganoff, D.; and McDonald, J. D.: A Collision-Selection Rule for an Efficient Particle-Simulation Method for Use on Vector Computers. Phys. Fluids A., vol. 2, issue 7, July 1990, pp. 1248-1259.
36. McDonald, J. D.; and Baganoff, D.: Vectorization of a Particle Simulation Method for Hypersonic Rarefied Flow. AIAA Paper 88-2735, June 1988.
37. McDonald, J. D.: Partial Simulation in a Multiprocessor Environment. RNR Technical Report RNR-91-003, Jan. 1991.
38. Dagum, L.: Three-Dimensional Direct Particle Simulation on the Connection Machine. RNR Technical Report RNR-91-022, Aug. 1991.
39. Feiereisen, W.; McDonald, J. D.; and Fallavollita, M.: Three-Dimensional Discrete Particle Simulation About the AFE Geometry. AIAA Paper 90-1778, 1990.
40. Haas, B. L.: Fundamentals of Chemistry Modeling Applicable to a Vectorized Particle Simulation. AIAA Paper 90-1749, 1990.
41. Wilmoth, R. G.: Application of a Parallel DSMC Method to Hypersonic Rarefied Flows. AIAA Paper 91-0772, Jan. 1991.
42. Molvik, G. A.; and Merkle, C. L.: A Set of Strongly Coupled, Upwind Algorithms for Computing Flows in Chemical Nonequilibrium. Presented at the AIAA 27th Aerospace Sciences Conference, Reno Nev., AIAA Paper 89-0199, Jan. 1989.
43. Lawrence, S. L.: Numerical Performance Estimates for a Generic Hypersonic Forebody. Presented at the AIAA 272nd Fluid Dynamics, Plasma Dynamics and Lasers Conference, Honolulu, Hawaii, AIAA Paper 91-1695, June 1991.
44. Buelow, P. E.; Tannehill, J. C.; Ievalts, J. O.; and Lawrence, S. L.: A Three-Dimensional Upwind Parabolized Navier-Stokes Code for Chemically Reacting Flows. AIAA J. Thermophysics and Heat Transfer, vol. 5, July-Sept. 1991, pp. 274-283.
45. Green, M. J. et al.: Application of CFD to a Generic Hypersonic Flight Research Study. Presented at the AIAA 31st Aerospace Sciences Meeting and Exhibit, Reno, Nev., AIAA Paper 93-0312, Jan. 1993.
46. Molvik, G. A.; Bowles, J. V.; and Huynh, L. C.: Analysis of a Hydrocarbon Scramjet with Augmented Preburning. Presented at the AIAA/SAE/ASME/ASEE 28th Joint Propulsion Conference and Exhibit, Nashville, Tenn., AIAA Paper 92-3425, July 1992.
47. Narayan, J. R.: Effect of Turbulent-Chemistry Interaction in Compressible Reacting Flows. AIAA Paper 94-2311, June 1994.
48. Drummond, J. P.: A Two-Dimensional Numerical Simulation of a Supersonic, Chemically Reacting Mixing Layer. NASA TM-4055, 1988.

49. Narayan, J. R.: Prediction of Turbulent Reacting Flows Related to Hypersonic Airbreathing Propulsion Systems. AIAA Paper 94-2948, June 1994.
50. Delaney, R. A.: Time-Marching Analysis of Steady Transonic Flow in Turbomachinery Cascades Using the Hopscotch Method. *Engineering for Power*, vol. 105, April 1983, pp. 272-279.
51. Davis, R. L.; Ron-Ho, Ni; and Carter, J. E.: Cascade Viscous Flow Analysis Using the Navier-Stokes Equations. AIAA Paper 86-0033, Jan. 1986.
52. Subramanian, S. V.; Bozzola, R.; and Povinelli, L. A.: Computation of Three-Dimensional, Rotational Flow Through Turbomachinery Blade Rows for Improved Aerodynamic Design Studies. ASME Paper No. 86-GT-26, 1986.
53. Weinberg, B. C.; Yang, R. J.; McDonald, H.; and Shamroth, S. J.: Calculations of Two- and Three-Dimensional Transonic Cascade Flow Fields Using the Navier-Stokes Equations. *J. Engineering for Gas Turbines and Power*, vol. 108, Jan. 1986, pp. 93-102.
54. Rai, M. M.; and Madavan, N. K.: Multi-Airfoil Navier-Stokes Simulations of Turbine Rotor-Stator Interaction. *J. Turbomachinery*, vol. 112, July 1990, pp. 377-384.
55. Rangwalla, A. A.; Madavan, N. K.; and Johnson, P. D.: Application of an Unsteady Navier-Stokes Solver to Transonic Turbine Design. Presented at the AIAA/SAE/ASME/ASEE 27th Joint Propulsion Conference, Sacramento, Calif., AIAA Paper 91-2468, June 1991.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE July 1994	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Computational Fluid Dynamics Uses in Fluid Dynamics/Aerodynamics Education			5. FUNDING NUMBERS 509-10	
6. AUTHOR(S) Terry L. Holst				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Ames Research Center Moffett Field, CA 94035-1000			8. PERFORMING ORGANIZATION REPORT NUMBER A-94105	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-108834	
11. SUPPLEMENTARY NOTES Point of Contact: Terry L. Holst, Ames Research Center, MS 258-1, Moffett Field, CA 94035-1000 (415) 604-6032				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Category - 02			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) The field of computational fluid dynamics (CFD) has advanced to the point where it can now be used for the purpose of fluid dynamics physics education. Because of the tremendous wealth of information available from numerical simulation, certain fundamental concepts can be efficiently communicated using an interactive graphical interrogation of the appropriate numerical simulation data base. In other situations, a large amount of aerodynamic information can be communicated to the student by interactive use of simple CFD tools on a workstation or even in a personal computer environment. The emphasis in this presentation is to discuss ideas for how this process might be implemented. Specific examples, taken from previous publications, will be used to highlight the presentation.				
14. SUBJECT TERMS Computational fluid dynamics, CFD, Aerodynamics, Numerical simulations			15. NUMBER OF PAGES 14	
			16. PRICE CODE A02	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT	